FEAMAC Example 3

***Description*:**

This FEAMAC example problem involves the analysis of a dogbone tension test specimen subjected to displacement controlled loading. The specimen geometry is shown in Fig. 12, and the quarter-symmetry finite element mesh, consisting of 432 C3D8 elements, is shown in Fig. 13. The material is again the same MAC/GMC material, a 33% volume fraction SiC/Ti composite with a simple 2×2 GMC repeating unit cell, with all elements and integration points utilizing MAC/GMC to provide the material constitutive response. A dogbone tensile specimen like the one modeled is intended to provide the material response in the gauge section where the strain is typically measured (via strain gauge, extensometer, or other method). Thus, the FEAMAC simulation should predict a stress-strain response in the specimen gauge section that is close to the response of the SiC/Ti material predicted in FEAMAC Example 1. By commenting and un-commenting lines in the ABAQUS and FEAMAC input files, simulations of both a longitudinally reinforced and transversely reinforced composite specimen are performed.

Required Files:

The following files should be placed in the ABAQUS working directory:

File	Purpose
`feamac_ex3.inp`	ABAQUS input file
`SiC-Ti_33.mac`	MAC/GMC input file describing the SiC/Ti composite material (with xy plots)
`SiC-Ti_33c.mac`	MAC/GMC input file describing the SiC/Ti composite material (without xy plots)
`feamac.for`	User-defined subroutines for FEAMAC

Execution:

This problem can be executed via the following command at the ABAQUS command line:

```
abaqus –j feamac_ex3 –user feamac interactive
```

The `–j` specification indicates the job name (i.e., ABAQUS input file name), while the `–user` specification indicates the file containing the FEAMAC user-defined subroutines. The `interactive` specification provides detailed information on the problem execution during the execution and is optional.

Output:

The output for this problem is written to the ABAQUS output database file `feamac_ex3.odb` for post-processing in ABAQUS/CAE, ABAQUS/Viewer, or other appropriate finite element post-processing software. In addition, MAC/GMC output is written to 8 ASCII files named `SIC-TI_33_macro_EL430_PT`*j*`.data`, where *j* is the integration point number in the ABAQUS finite element model. This file contains the macro (repeating unit cell) level output specified in the MAC/GMC input file. Element number 430 is assigned the material `SiC-Ti_33.mac`, which has xy plot data output specified, while the remaining elements of the model are assigned the material `SiC-Ti_33c.mac`, which does not have xy plot data output specified. The xy plot data is thus only written for element number 430, which is located at the middle of the dogbone specimen gauge section. Finally, a MAC/GMC output file is written for each MAC/GMC input file employed in the ABAQUS model. In this case these files are `SiC-Ti_33.out` and `SiC-Ti_33c.out`. It contains an echo of the MAC/GMC input file data and results in the form of effective properties. If an error is found in the MAC/GMC input, a message describing the error will be written to this output file.

Results:

Stress-strain results are plotted in Fig. 14. For the FEAMAC dogbone cases, the stress-strain curves are plotted from the data in the `SIC-TI_33_macro_EL430_PT`*j*`.data` files (which are nearly identical for the different integration points). This element is located near the middle of the specimen gauge sections (See Fig. 13), and thus is intended to represent the response of the material as would be measured via a strain gauge or extensometer. As shown if Fig. 14, the results match closely with stand-alone MAC/GMC analyses of the SiC/Ti composite material. The slight deviations are due to the difference in control mode (displacement control for FEAMAC, strain control for MAC/GMC), which results in a difference in strain rate. The von Mises stress and equivalent plastic strain fields for the longitudinal composite specimen are plotted in Figs. 15 and 16 at the end of the applied loading. These figures show that, while the fields are close to constant in the middle of the gauge section, a concentration arises in the gauge section reduction region.

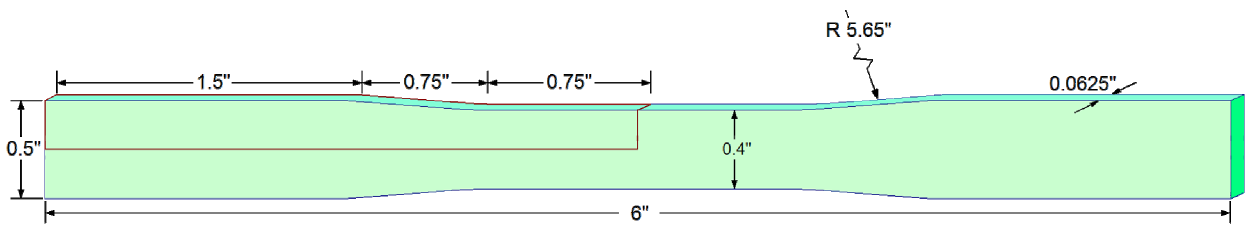

Fig. 12. Geometry of the dogbone specimen analyzed in FEAMAC Example 3.

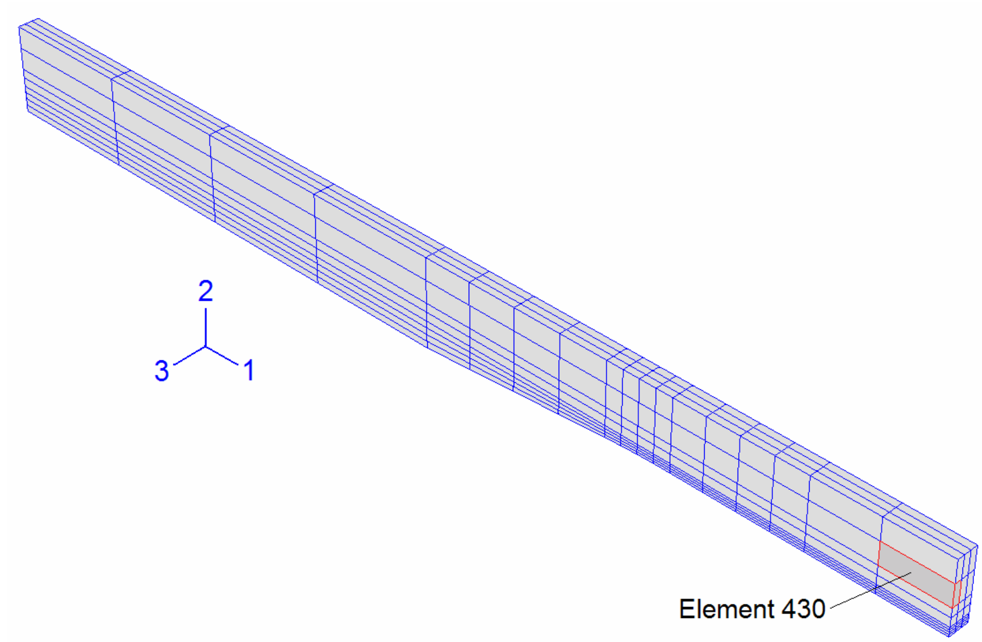

Fig 13. Quarter-symmetry mesh of the dogbone specimen consisting of 432 C3D8 elements.

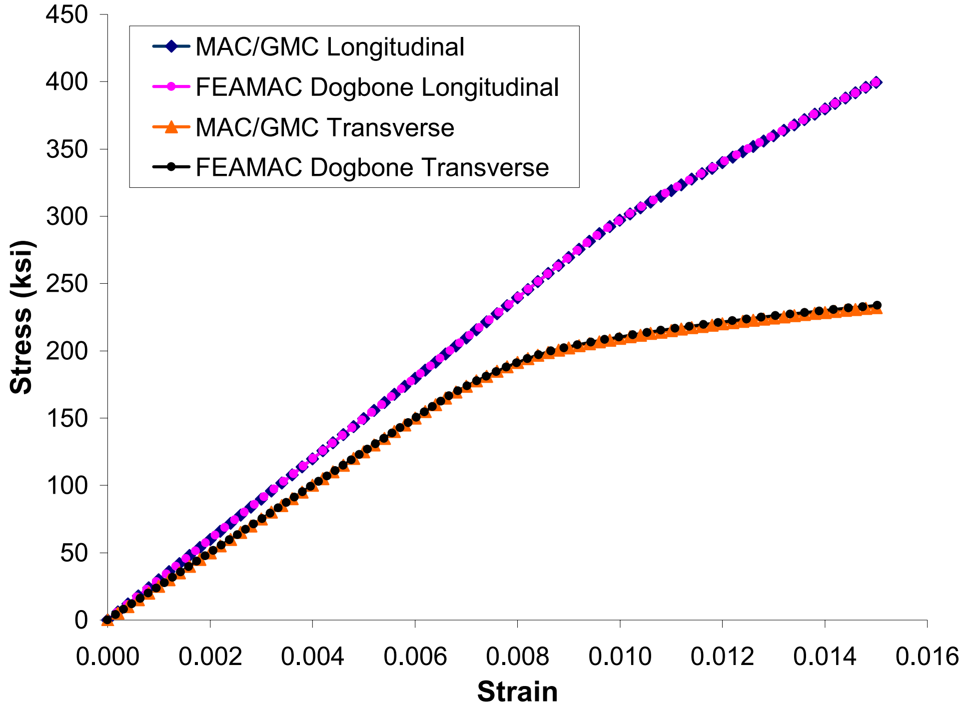

Fig. 14. Comparison of longitudinal and transverse stress-strain curves for a 33% SiC/Ti composite dogbone specimen simulated by FEAMAC and stand-alone MAC/GMC simulations for the material.

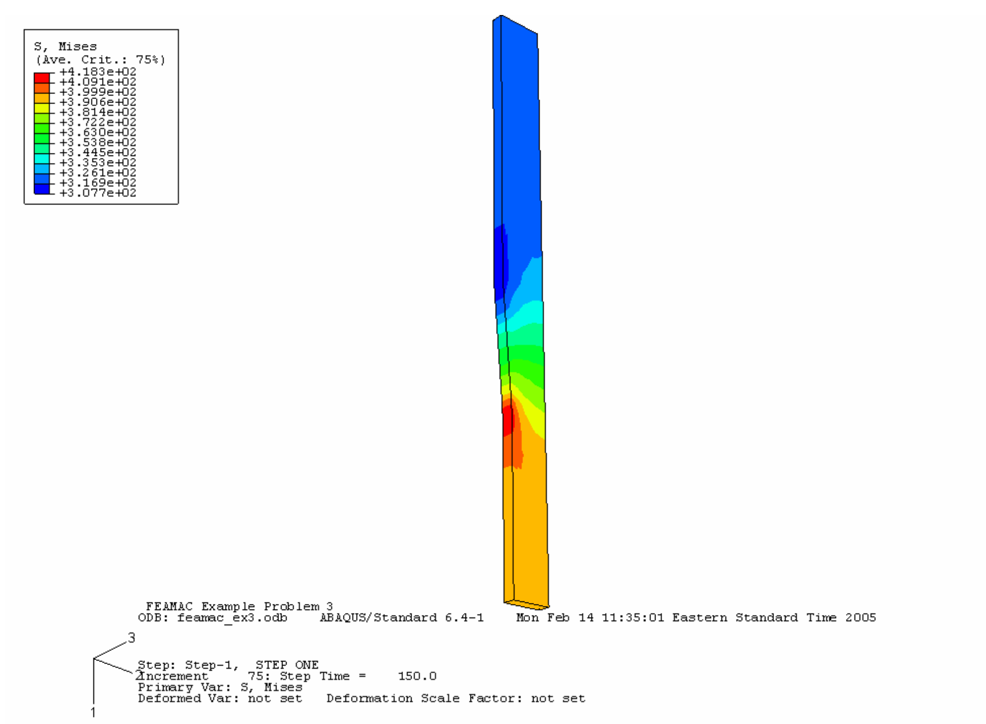

Fig. 15. von Mises stress field (ksi) for a longitudinally reinforced 33% SiC/Ti composite dogbone specimen simulated by FEAMAC.

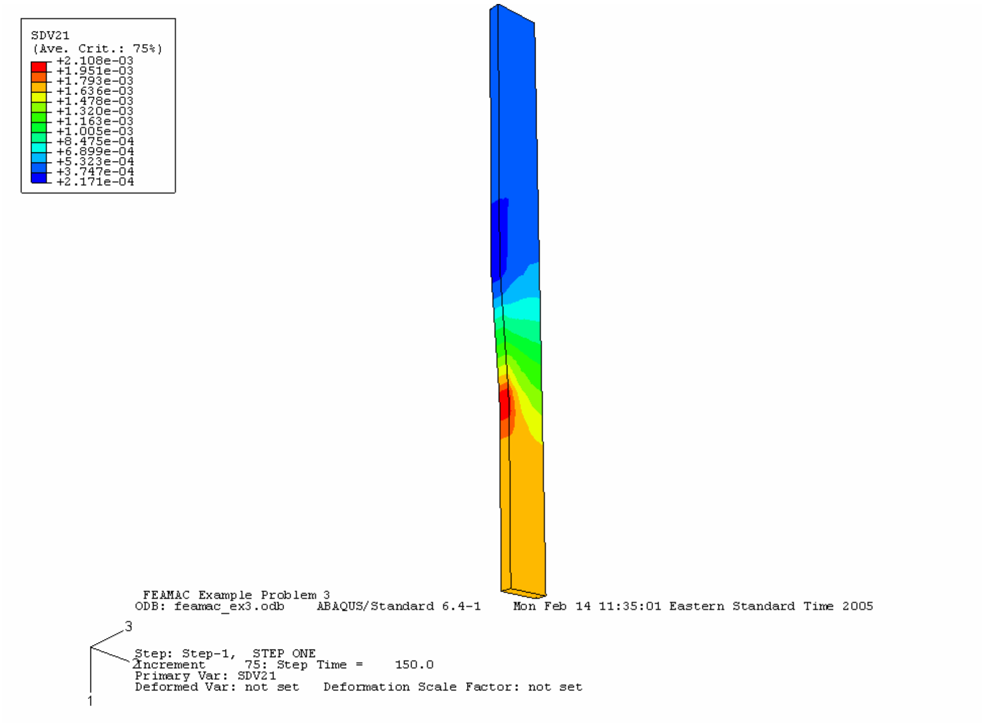

Fig. 16. Equivalent plastic strain field for a longitudinally reinforced 33% SiC/Ti composite dogbone specimen simulated by FEAMAC.